

ECE322L - Lab 1

Circuit Simulation

Roger Holten, David Kirby, Landon Schmucker

Goal

Explore and gain understanding of Pspice and Multisim software.

Software needed

- PSpice and MultiSim

The Lab

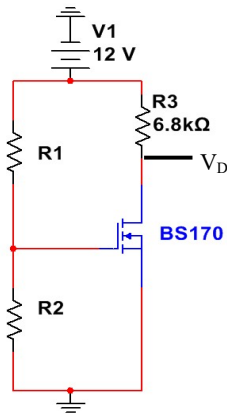


Figure 1: Circuit Diagram.

Part 1 – Hand Calculations

Let $K'_n = 1.825 \text{ mA/V}^2$, $(W/L) = 1$, $V_{TN} = 1 \text{ V}$, and $R1 + R2 = 100\text{K}$. Bias the transistor such that half the supply voltage is across the transistor.

1. Find I_{DQ} such that half the supply voltage is across the transistor
2. Find R_1 and R_2 through hand calculations.
3. Solve for V_{DSQ} and V_{GSQ} .

Use the following table to keep track of the measurements.

	Hand Calc	PSPICE	MultiSim
I_{DQ}	0.8824mA	0.876mA	0.876mA
V_{DSQ}	6V	6.04V	6.04V
V_{GSQ}	1.9833V	1.98V	1.98V
R_1	83.472k Ω	N/A	N/A
R_2	16.528k Ω	N/A	N/A

LAB 01 ECE 322-L 29-Jan

Hand Calculations:

$$K'_n = 1.825 \frac{\text{mA}}{\text{V}^2} \quad \frac{W}{L} = 1 \quad V_{TN} = 1\text{V}$$

$$R_1 + R_2 = 100\text{k}\Omega \quad R_3 = 6.8\text{k}\Omega \quad V_{DD} = 12\text{V}$$

$$I_D = \frac{1}{2} K'_n \left(\frac{W}{L}\right) (V_{GS} - V_{TN})^2 \quad V_{GS} = V_{DD} \frac{R_2}{R_1 + R_2}$$

$$V_{DS} = V_{DD} - R_D I_D \quad V_{GS} = 12\text{V} \left(\frac{R_2}{100\text{k}\Omega}\right)$$

$$6\text{V} = 12\text{V} - 6.8\text{k}\Omega I_D$$

$$6.8\text{k}\Omega I_D = 6\text{V}$$

$$I_{DQ} = 0.8824\text{mA}$$

$$0.8824\text{mA} = \frac{1}{2} (1.825 \frac{\text{mA}}{\text{V}^2}) (1) (V_{GS} - 1\text{V})^2$$

$$V_{GSQ} = 1.9833\text{V}$$

$$R_2 = 16.528\text{k}\Omega \quad R_1 = 83.472\text{k}\Omega$$

$$V_{DSQ} = V_{DD} - R_D I_D$$

$$= 6\text{V}$$

Part 2 – Modeling the Circuit in Pspice

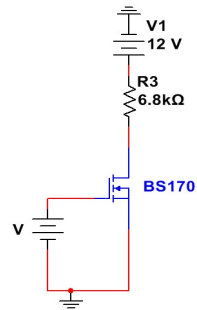
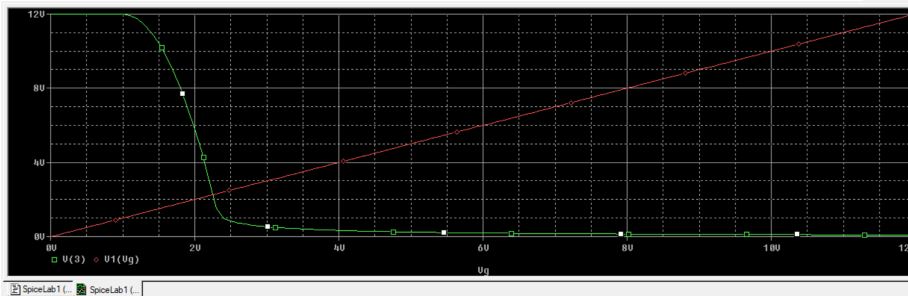


Fig 2: PSPICE circuit diagram

1. Create the circuit in PSPICE

```
V1 1 0 12
*Gate Voltage, we will use this to sweep for VGSQ
Vg 2 0 1.98
*Drain Resistor < + node> < -node> <Resistance>
Rd 1 3 6.8k
*label <drain> <gate> <source> <bulk> <.model name>
M1 3 2 0 0 ntype
.model ntype nmos vto=1 kp=1.825m l=1.0u w=1.0u
*.dc VG 0 4 0.01
.probe
*operating point
.probe
.op
.end
```

2. Provide a graph with V_D as a function of V_G in your report.



```

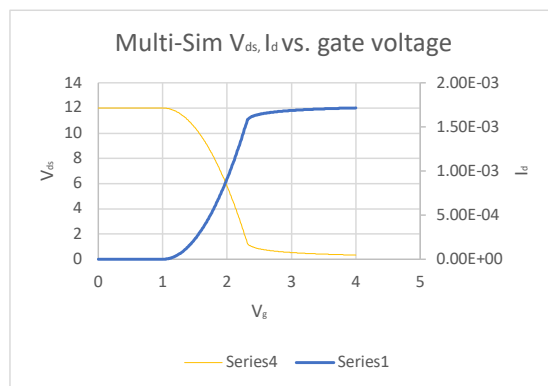
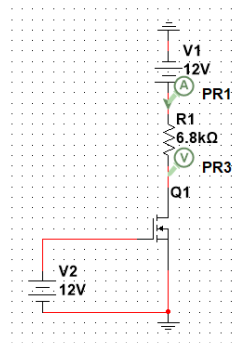
**** MOSFETS

NAME      M1
MODEL     ntype
ID        8.76E-04
VGS       1.98E+00
VDS       6.04E+00
VBS       0.00E+00
VTH       1.00E+00
VDSAT     9.80E-01

```

Part 3 - MultiSim

1. Created the circuit shown in figure 2.
2. In MultiSim we can place probes which can help with quick analysis and setting up simulations.



3. Compare the differences of the quiescent values obtained through PSPICE and hand calculations. What could cause the differences?

The differences between quiescent values of PSPICE and hand calculations is minimal and most likely just due to rounding in the PSPICE software. Another possible issue is in the PSPICE we only did steps of 0.01V for the DC Sweep which would cause the small rounding error.

4. Which simulation software do you prefer? Why? (not limited to Pspice and Multisim)

We personally prefer Pspice due to its ability to work with netlist and visual circuit creation. It also tends to be a little easier to do multiple simulations at once as you can just add them to the end of the netlist. It also is easier to edit the components since you can quickly change the netlist.

Commented [LS1]: